



## 9 SMT Board Assembly Process Recommendations

9	SMT Board Assembly Process Recommendations .....	9-1
9.1	Introduction.....	9-2
9.2	Solder Paste Printing.....	9-2
9.3	Component Placement .....	9-2
9.4	Reflow Soldering.....	9-3
9.4.1	Pb-free vs SnPb Reflow Soldering .....	9-3
9.4.2	Reflow Profile Development Considerations .....	9-3
9.4.2.1	Reflow Profile Board Preparation .....	9-4
9.4.2.2	Minimum Solder Joint Peak Temperature .....	9-5
9.4.2.3	Maximum Solder Joint Peak Temperature .....	9-5
9.4.2.4	Time Above (Initial) Melting Point .....	9-5
9.4.2.5	Rising and Falling Ramp Rate .....	9-6
9.4.2.6	Reflow Equipment .....	9-6
9.4.2.7	Reflow Atmosphere .....	9-6
9.4.2.8	Board Warpage .....	9-7
9.4.2.9	Double Sided SMT Board Assembly .....	9-7
9.4.3	Sample Reflow Parameters.....	9-7
9.4.4	Sample Reflow Profiles .....	9-8
9.4.4.1	Sample Desktop Reflow Profile .....	9-9
9.4.4.2	Sample Mobile Reflow Profile .....	9-9
9.4.4.3	Sample Server Reflow Profile .....	9-10
9.5	Rework.....	9-10
9.5.1	Pb-free vs SnPb Rework .....	9-10
9.5.2	Risk of SnPb and Pb-free Mixing.....	9-10
9.5.3	Rework Profile Board Preparation.....	9-10
9.5.4	Pad Cleanup After Component Removal .....	9-11
9.5.5	Paste Printing Methods at Rework .....	9-12
9.5.6	Re-balling BGAs Not Recommended .....	9-13
9.5.7	Sample Rework Parameters.....	9-14
9.5.8	Rework Profile Development .....	9-15
9.5.9	Sample Rework Profiles .....	9-16
9.5.9.1	Sample BGA Rework Profile .....	9-16
9.5.9.2	Sample Socket Rework Profile.....	9-17



## **9.1 Introduction**

This chapter addresses the surface mount technology (SMT) board assembly process for reflow soldering SMT components to boards, as well as rework soldering for removing and replacing individual components on already-assembled boards.

The information in this document is for reference only. Manufacturing processes are unique, and may require unique solutions to ensure acceptable levels of quality, reliability, and manufacturing yield. Due to differences in equipment and materials, customer-specific process parameter development and validation is required.

## **9.2 Solder Paste Printing**

Standard tin-lead (SnPb) solder paste alloy is composed of 63% tin (by weight) and 37% lead, which is commonly expressed as 63Sn/37Pb or Sn/37Pb, a eutectic composition that melts at 183C.

Although there are a number of lead free (Pb-free) alloys, the most commonly used compositions contain tin, silver, and copper, commonly expressed as SAC, for SnAgCu. Within SAC solders, by far the most common usage is Sn/3Ag/0.5Cu, a near-eutectic which melts between 217°C and 220C.

Apertures sizes can be 1:1 with pad size, but certain parts may require reduced apertures to reduce solder ball defects. Larger pads may benefit from crosshatched openings, to reduce the amount of paste applied, and to control scavenging.

Pb-free solder paste may spread less during reflow than SnPb paste, potentially leaving extremities of pads unsoldered. Although full pad coverage by solder is not a requirement of IPC-A-610, some customers prefer to enlarge apertures to ensure that pads are covered.

Using a metal squeegee reduces scavenging and provides more consistent printed paste volume.

Equipment used to print SnPb paste can be used, without modification, to print Pb-free paste.

Process parameters (such as squeegee speed, pressure, and separation speed) need to be optimized for the specific solder paste used.

## **9.3 Component Placement**

Pick and place machines used for SnPb boards can be used for Pb-free boards as well. Adjustments to lighting and vision algorithms may be required because of the slightly different appearance of some Pb-free solders compared to SnPb. Specifically, SnPb solders can have a grainy and dull (less shiny) appearance. This applies only when front-side lighting is used instead of backside (outline) lighting. Front-side lighting is often used for ball recognition on BGAs.



## **9.4 Reflow Soldering**

In reflow soldering, the solder paste must be heated sufficiently above its melting point and become completely molten, in order to melt the balls of BGA components, causing them to collapse and form reliable joints. In the case of components with leads, the solder paste must wet the plating on component leads to form the desired heel and toe fillets.

Solder joint formation depends on temperature and time which are reflected in the reflow profile. In leaded devices the volume of solder paste on the land is significantly greater than the plated solder volume on the component lead and is the key contributor to joint formation. However in BGAs the balls on the component are the main contributor to the solder volume of the joint. In both cases, the paste volume applied is critical to the formation of the joint.

There is no one best reflow profile for all board assemblies. Ideally, a reflow profile must be characterized for each board assembly using thermocouples at multiple locations on and around the device. The solder paste type, component and board thermal sensitivity must be considered in reflow profile development.

### **9.4.1 Pb-free vs SnPb Reflow Soldering**

Compared to SnPb reflow, Pb-free reflow requires higher temperatures, due to the higher melting range of typical Pb-free solders. While typical tin-lead solder (Sn/37Pb) has a single melting point of 183C, typical Pb-free solder such as SAC305 (Sn/3Ag/0.5Cu) has a much higher initial melting point of 217°C and a final melting point of 220C.

In addition to having higher reflow temperatures, Pb-free reflow soldering also requires a narrower temperature range, in order to produce reliable joints, without damaging components. Maintaining this narrower range could require new reflow ovens, depending on number of zones and degree of control in ovens formerly used for SnPb soldering.

Because of additional oxidation that occurs at higher temperatures, an inert reflow atmosphere (nitrogen) may be beneficial for Pb-free reflow soldering.

Of course, higher temperatures drive the need for all Pb-free components to be rated to higher temperatures.

Finally, these temperatures can also cause greater warpage in PCBs, and in some cases, may require alternate PCB materials, or carrier fixtures during reflow.

### **9.4.2 Reflow Profile Development Considerations**

Each customer should develop their own reflow profile and oven settings, appropriate to their materials, equipment, and products. As a starting point, this chapter contains considerations and recommendations for reflow solder parameters. Because some reflow parameters differ with solder paste formulation (even if they have the same metal composition), the profile envelope recommended by the solder paste manufacturer should be considered.



### 9.4.2.1 Reflow Profile Board Preparation

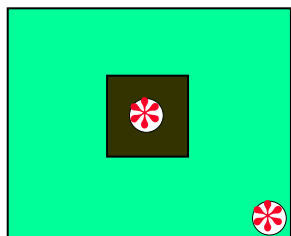
Reflow profile measurement is a vital part of setting up reflow solder conditions. The measurements are typically carried out using thermocouples attached to a high temperature resistant recording device which travels through the reflow oven with the PCB under test. Special care must be taken to ensure proper placement of thermocouples to accurately measure temperature at the desired locations.

Unless stated otherwise, all temperatures in this chapter are measured at solder joints, rather than at components bodies, PCB surface, or air around components. This provides the best repeatability and accuracy.

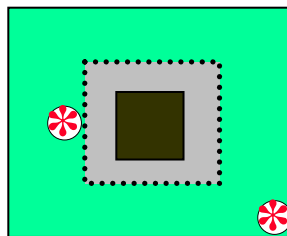
Thermocouples (TCs) for solder joints should be placed in joints expected to be the hottest and coolest, so that the range of peak temperatures for all components on the board can be confirmed to be within specifications. The hottest joints on a board are typically on small passive components, so one of these should be monitored for peak temperature on the profile board. The coolest joints on a board are typically large BGAs and sockets. A TC should be used in a joint at one corner of the component, and in a joint at the center of the part, or as near to the center of the part as possible. Sockets with actuating mechanisms may require an additional TC at a joint near the mechanism, if its mass could make that area harder to heat.

In addition to solder joints, component body temperature, measured at top center or as close as possible, may also need to be monitored, to avoid exceeding the body temperature spec of the part.

Here are examples of TC locations for reflow profiling on BGAs or sockets, for both fully populated arrays and partially populated arrays (no balls in the center area of the part).



Fully populated array.



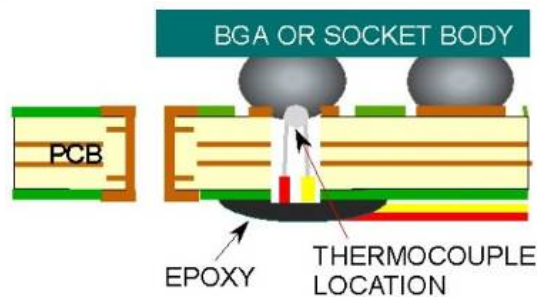
Partially populated array.

 = Location of TC in solder joint.

A topside TC to monitor body temp could also be placed at the center of each part.

Here is a method for placing TCs to measure SMT joint temperatures:

- Before the component is soldered to the board, drill a small hole through the pad of the joint to be measured.
- Insert the thermocouple from the bottom of the board.
- Hold the thermocouple tip flush with the top surface of the board.
- Apply epoxy from the bottom side of the board to keep the thermocouple in this position, where it will be in contact with the joint, but not interfere with paste printing.
- Print solder paste, place components, and reflow the board.



#### 9.4.2.2 Minimum Solder Joint Peak Temperature

With SAC305 or SAC405 Pb-free solder paste, the coolest joints on a board should generally reach at least 228°C, preferably 230°C. For BGAs, this applies to ball alloys SAC305 or SAC405. For SAC305/405, 228°C represents at least 11°C superheating above the initial melting point (217°C). 230°C represents 13°C superheating. Temperatures lower than these can result in joints that are not fully formed, or in reduced reliability.

Components with other Pb-free ball alloys, such as SnAg or SAC105, may require higher minimum peak temperatures to form reliable joints.

#### 9.4.2.3 Maximum Solder Joint Peak Temperature

250°C is recommended as the maximum temperature for all solder joints on the board, except for components with temperature ratings lower than 250°C. If maximum solder joint temperatures exceed 250°C, PCB damage such as delamination and warpage may result when standard FR4 ( $T_g = 130^\circ\text{C}$ ) material is used. Higher  $T_g$  material is not necessarily more resistant to this damage, and must be tested for compatibility.

Components are typically rated as per J-STD-020C (or later), based on their package thickness and volume. Although Intel BGAs are generally rated at 260°C, other components, especially large ones, may be rated at 250°C or 245°C. This means that 250°C may not be usable as the max joint temperature for some components; a lower temperature may be required. Since larger parts normally reach lower maximum temperatures during reflow than smaller parts due to the physics of heat transfer, keeping them below their ratings may not be difficult, as long as 250°C is used as the maximum for the joints of smaller parts.

#### 9.4.2.4 Time Above (Initial) Melting Point

The length of time that joints spend above the min peak temp is also an important factor for solder joint reliability. Intel recommends that this time be measured from the time a joint goes above the initial melting point of the alloy (217°C for SAC305/405), until it goes below it during cooling.

Time Above Liquidus, or TAL, is often used to describe this time. But technically speaking, 217°C is the *solidus* temperature of SAC305, the point at which the solder becomes *fully solid during cooling*. 220°C is the *liquidus* temperature, the point at which it becomes *fully liquid during heating*. Between these two temperatures, the solder is partially molten and partially solid.



Unfortunately, common usage in the lead free industry has often incorrectly used the term ‘liquidus’ to refer to the initial melting point, rather than the final melting point. This is probably a carry-over from SnPb soldering, where ‘liquidus’ was used correctly, since liquidus and solidus are both the same temperature (183°C). In order to avoid confusion, this document will avoid the use of ‘liquidus’ and ‘Time Above Liquidus (TAL)’. Reflow time will be stated as Time Above 217°C (TA217), rather than TAL.

Intel recommends that TA217 of 40-90 seconds be used for SAC305 or 405 solder paste and balls. With large or massive boards, an exception may be required, allowing up to 120 seconds above 217°C.

#### **9.4.2.5 Rising and Falling Ramp Rate**

To avoid component damage, manufacturers often recommend that rate of temperature increase during heating (Rising Ramp Rate) and cooling (Falling Ramp Rate) be kept below 3°C/sec. This applies throughout the heating and cooling process.

During cooling from peak temperature down to 205°C, Intel also recommends that a *minimum* ramp rate be used. Solder joints with cooling rates of 1°C/sec or greater are characterized by finer microstructure features. Literature studies indicate that this is better for long term reliability. Faster cooling rates also inhibit growth of intermetallic compounds in the bulk solder.

#### **9.4.2.6 Reflow Equipment**

The peak temperature envelope is typically narrower for Pb-free reflow than for SnPb reflow. Although ovens designed for SnPb soldering can generally reach the higher temperatures required for Pb-free soldering, they may not be able to produce the narrower temperature profile, at least not without significantly lengthening reflow time.

Producing and controlling a narrower temperature range, while maintaining production speeds, typically relates to the number of heating zones in the oven. Assemblers with reflow equipment with greater temperature control (i.e. greater numbers of zones) will be better positioned to meet the tighter Pb-free process envelope requirements, particularly for larger, more complex boards.

#### **9.4.2.7 Reflow Atmosphere**

Reflow soldering in an inert atmosphere, such as nitrogen, reduces the amount of solder and pad oxidation that occurs during soldering. This can improve the quality and appearance of SMT joints, and have a positive impact on hole fill at wave solder and on contact resistance during test, especially with PCBs using OSP surface finish.

However, other measures can often be taken to achieve similar results without using nitrogen at reflow. Examples include:

- Solder paste selection.
- PCB surface finish selection.
- Printing solder paste on test pads rather than leaving them exposed during reflow.
- Test probe head style selection.



Examples specific to wave soldered boards include:

- Wave flux selection.
- Wave flux application method.
- Wave flux volume applied.
- Wave flux distribution and depth of penetration in holes.
- Wave solder parameters, such as:
  - Preheat configuration.
  - Preheat profile.
  - Wave solder alloy selection.
  - Wave solder pot temperature.
  - Solder wave dynamics.

#### 9.4.2.8 Board Warpage

Because of the higher temperatures required for Pb-free assembly, boards may sag and warp more than during SnPb assembly. This is particularly noticeable on thin PCBs, such as those used in mobile applications. Although there is no common industry specification for the warpage of assembled boards (only for bare boards), board assemblers may prefer to reduce warpage.

- A picture-frame style pallet, with hold downs, can be used to support the board on all four sides.
- Channel-type support rails can be attached to the leading and trailing edge of the board, after solder printing but prior to reflow.
- Oven manufacturers offer various center support mechanisms, such as an adjustable center support wire or chain, but this imposes a placement stay-out zone on the design.

Warpage amount varies with PCB size/thickness/laminate, number of reflow cycles, and warpage control method. In experiments at Intel, warpage of 1.0 and 1.2mm thick boards did not cause any performance problems.

#### 9.4.2.9 Double Sided SMT Board Assembly

Both primary and secondary side reflow profiles should meet the same target specification. Because the board assembly has greater thermal mass during second reflow, different oven settings may be required first and second reflow, in order to meet the same target profile. This requires two profile boards to simulate actual board configuration (thermal mass) during each respective reflow.

#### 9.4.3 Sample Reflow Parameters

Solder paste	No-clean, flux class ROL0 per J-STD-004. Alloy Sn/3Ag/0.5Cu or Sn/4Ag/0.5Cu. Metal content 89%.
Soak	Paste dependent: Consult paste manufacturer.
Rising Ramp Rate	Maximum 3°C per second.
Falling Ramp Rate	Maximum 3°C per second. Minimum 1°C per second, from peak temp down to 205°C.
Time Above 217°C	Prefer 40-90 seconds above 217°C. Min 40 seconds, Max 120 seconds
Minimum solder joint peak temp	228°C, if <i>all</i> BGA balls on board are SnAgCu 230°C, if <i>any</i> BGA balls on board are SnAg

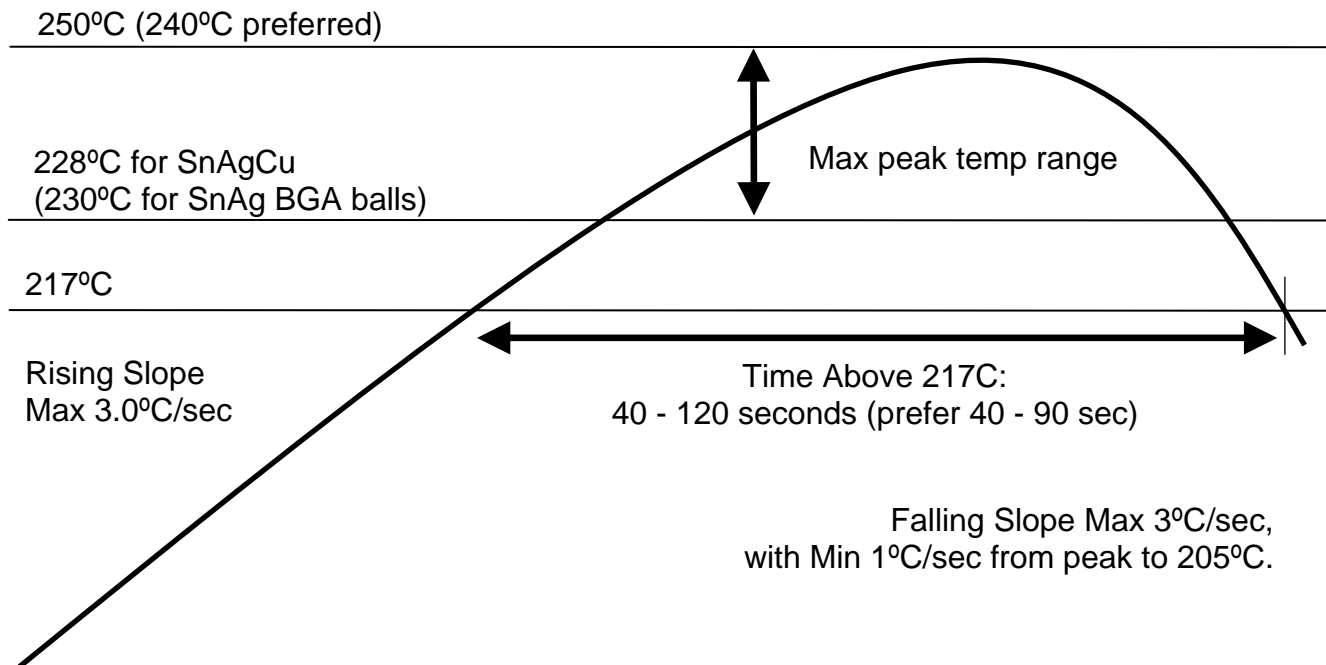




Maximum solder joint peak temp	250°C, except as limited by components with lower temperature ratings. Preferred joint temp max 240C.
Maximum body temp	Not to exceed manufacturer specification. If no spec, then not to exceed temps as per J-STD-020C.
Oven type	Forced convection

- Sample process applies to all types of SMT components on the board, not just the BGAs.
- All temperatures are measured with thermocouples inside solder joints, for better accuracy
- Max temp applies to the hottest joint on the board, typically a joint of a small passive device.
- Reference process applies to all PCBs with nominal thickness .040” to .077” (1.02 to 1.96mm), and to PCBs .078” to .093” (2.0 to 2.36 mm) with large active devices on one side only. Thick PCBs with thermally massive parts on both sides may require adjustments to Peak Temp and TA217.

Here is a graphical representation of information in the table above.



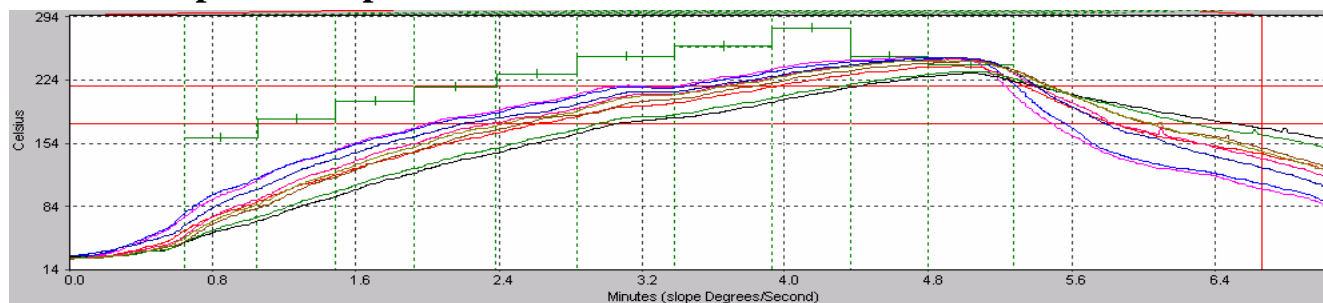
#### 9.4.4 Sample Reflow Profiles

Here are sample reflow profiles for certain Pb-free desktop, mobile, and server boards. These are not meant to indicate reflow requirements, merely to illustrate typical reflow profiles.



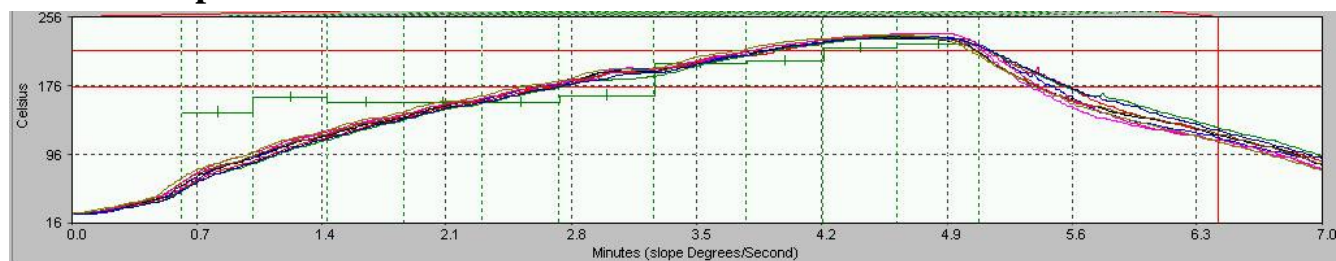


### 9.4.4.1 Sample Desktop Reflow Profile



	Peak	Max Rising Slope	Max Falling Slope	Time Between 175/217	Total Time Above 217
1 U1F2	247.8	2.51	-3.17	76.36	117.28
2 0603 CAP	247.3	2.85	-2.96	85.17	109.76
3 SKT CORNER	238.7	2.08	-1.67	80.07	82.12
4 SKT CENTER	230.6	1.37	-0.84	81.29	57.94
5 SKT LEVER	232.9	1.53	-1.01	81.88	65.62
6 MCH CENTER	243.5	2.14	-1.54	78.74	97.92
7 MCH CORNER	246.5	2.34	-2.07	77.22	104.09
8 U4A1	247.3	2.31	-2.24	84.23	107.56
9 U5G1 ICH	245.0	2.07	-1.55	75.10	106.78
TC Range	17.2	1.48	2.33	10.06	59.34

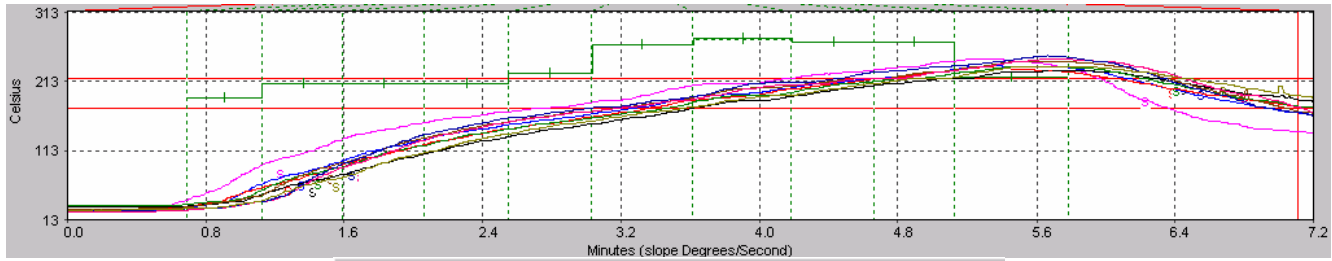
### 9.4.4.2 Sample Mobile Reflow Profile



	Peak	Max Rising Slope	Time Between 175/217	Total Time Above 217
1 MCH3	233.3	2.42	77.14	71.74
2 MCH3	233.6	2.24	76.31	74.15
3 CPU1	232.6	2.04	76.10	70.06
4 CPU1	231.2	2.35	76.55	71.18
5 MCH1	232.1	1.95	70.29	72.66
6 MCH1	233.7	2.26	70.34	74.69
7 ICH1	236.6	2.43	71.24	79.40
8 ICH1	233.1	2.09	70.05	74.36
9 EDEN	234.9	2.49	72.25	77.87



### 9.4.4.3 Sample Server Reflow Profile



	Peak	Max Rising Slope	Max Slope	Time Between 175/217	Total Time Above 217
1 Q8A2	246.3	1.67	1.67	81.60	109.04
2 J8K1 COR	235.9	1.46	1.46	80.75	89.76
3 J2K1 CNT	233.1	1.26	1.26	79.33	83.02
4 U5G2 CNT	229.0	1.11	1.11	86.70	73.78
5 U5G2 COR	233.6	1.34	1.34	85.55	80.43
6 U15	242.2	1.56	1.56	92.16	99.11
7 U3 CNT	245.5	1.72	1.72	83.43	112.21
8 C3A16	249.4	1.88	1.88	84.57	113.53
9 J100 CNT	234.6	1.29	1.29	78.18	96.43

## 9.5 Rework

### 9.5.1 Pb-free vs SnPb Rework

Rework must provide higher temperatures for Pb-free solder. Greater temperature profile control is required, which may require different nozzles or equipment replacement. Rework can be the most difficult module to develop for Pb-free. Rework (both SMT and Through Hole) on thick PCBs is especially challenging.

### 9.5.2 Risk of SnPb and Pb-free Mixing

During the transition from SnPb to Pb-free assembly, rework is a potential area for inadvertently mixing Pb-free with SnPb boards. Dedicate separate areas, tools, and equipment to Pb-free and SnPb. Clearly identify SnPb and Pb-free work areas. Ensure that boards are clearly identifiable as Pb-free or SnPb.

### 9.5.3 Rework Profile Board Preparation

Because a rework profile is developed for a single component at a time, rather than the entire board, each component can have many thermocouples (TCs) on it, rather than the one or two locations used on reflow profile boards.

TCs should initially be located at:

- Solder joints at all four corners of the hot air reworked component.
- A solder joint at the center of the part, or as near to the center as possible, to represent the coolest joints on the component.

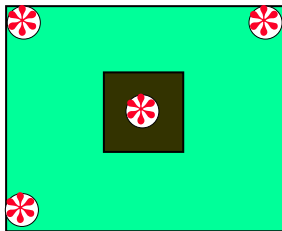


- Sockets with actuating mechanisms may require an additional TC at a joint near the mechanism, if its mass could make that area harder to heat.

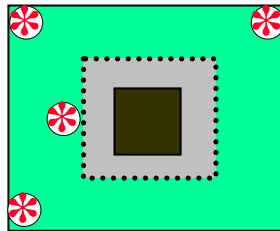
After developing the initial profile, place an additional TC at the topside location corresponding to the TC with the hottest joint temperature.

- Because of the nature of hot air rework, and the variety of nozzle designs, there may be a significant temperature gradient across the part during rework.
- Therefore, monitoring body temperature with a TC only in the center may not represent what the rest of the body is exposed to.
- Use this topside TC to confirm that component body temp is not exceeding its max rating. Adjust profile if needed.


Here are examples of TC locations for rework profiling on BGAs or sockets, for both fully populated arrays and partially populated arrays (no balls in the center area of the part).



Fully populated array.



Partially populated array.

 = Location of TC in solder joint.

A topside TC to monitor body temp would be also placed at one of these points, depending on which area has the highest joint temps.

TCs to measure joint temperatures are installed through holes in the board, using the same method described earlier for reflow profile boards.

#### 9.5.4 Pad Cleanup After Component Removal

While wicking solder off of pads:

- Always clip off the used portion of the wick; it behaves as a heat sink.
- Apply liquid flux to the wick, to minimize sticking of the wick to the pads.
- Place the soldering iron on the solder wick off to the edge of the pads being soldered, to heat iron tip and wick prior to desoldering.
- Do not let the solder iron or wick freeze on pads, to prevent pad lift.
- Do not lift the iron or wick up and down on the pads.
- Apply very light pressure, similar to writing with a pencil. Soldering is achieved by temperature difference, not by tip pressure.
- Apply heat for 2 to 3 seconds after solder melts. Total contact time may be 6-7 seconds. Excess heating causes solder brittleness and may lift pads.
- Move the soldering iron in the same direction with each stroke, rather than going back and forth. Going back and forth overheats pads at the ends of the row, increasing potential for damage.

Pb-free hand solder may require soldering iron tips hotter than used for SnPb rework. Hotter tips allow rework at a pace similar to SnPb rework. Without hotter tips, desoldering and resoldering is slower.



However, with hotter tips, caution must be used to prevent pad lift. If Pb-free tips are not available, Pb may be purged from standard SnPb tips by repeatedly flooding with Pb-free solder and then cleaning.

### 9.5.5 Paste Printing Methods at Rework

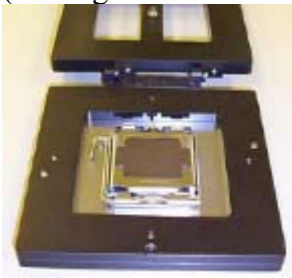
When new BGAs are installed at Rework, after pad cleanup, additional solder paste may not be required. The BGA ball can provide enough solder for a good joint. When new processor sockets are installed at Rework, additional paste is generally needed to ensure that good joints are formed, due to greater coplanarity differences.

Because of the difficulties of placing, aligning, and printing with a mini-stencil on the assembled PCB, a method is available that prints paste directly onto the socket BGA balls, rather than onto the PCB, as follows:

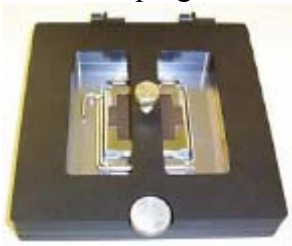
1. A stencil is inserted into the ball printing jig.



2. The part is hand placed with balls resting in stencil apertures.  
(An alignment frame is used, but not shown.)



3. A clamping frame holds the part in place for later operations.





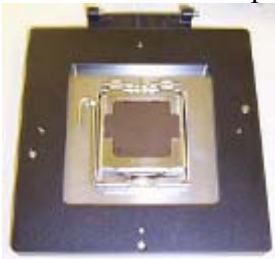
4. The jig is inverted, and paste is applied over the apertures.



5. A mini squeegee prints paste onto the balls, and removes excess paste.



6. The jig is inverted once more, the clamping frame is opened, and the part is removed by the rework machine's vacuum pick for placement onto the board.



Resulting side view of paste-printed balls.



### 9.5.6 Re-balling BGAs Not Recommended

Removed BGAs should be discarded. The re-balling process (placing new balls on removed BGAs, so that they can be re-used) is not recommended, for these reasons:

- Many BGAs (including Intel BGAs) are rated for three soldering cycles.
  - Re-balling exposes BGAs to more than three soldering cycles.
  - (1) Initial installation, (2) Rework/removal, (3) Ball attach, and (4) Final installation.
  - BGAs used on double sided boards could have even more soldering cycles.

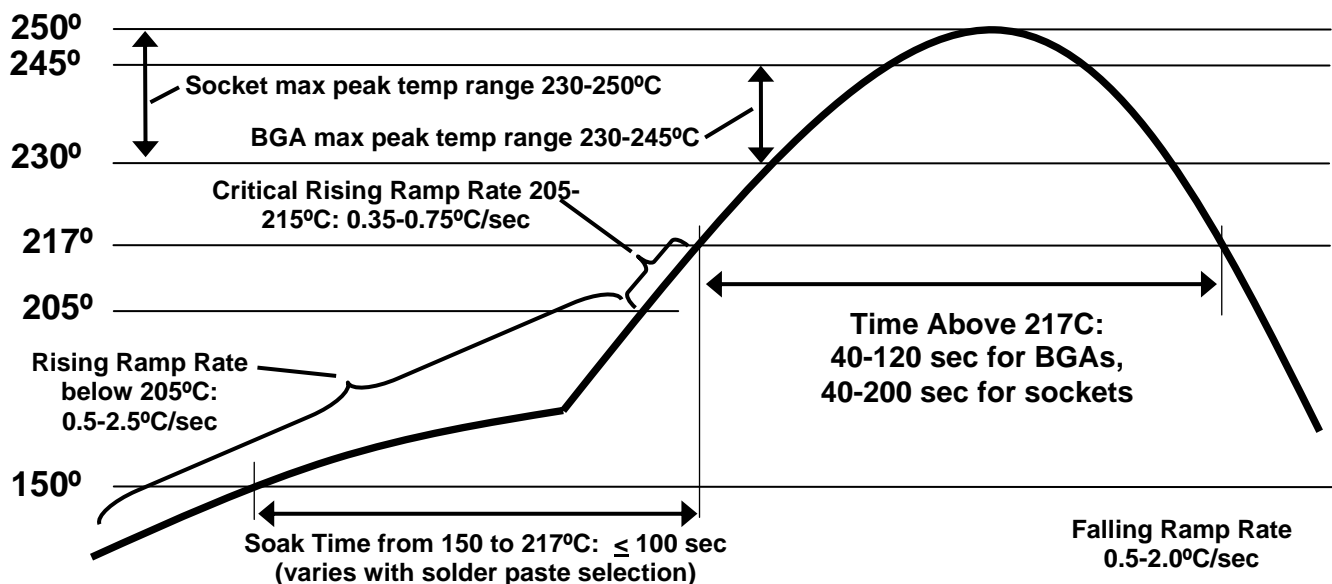


- Exposing BGAs to more than three soldering cycles may void the manufacturer's warranty (including Intel's).
- The intermetallic compound (IMC) layer, on the PCB and on the package, gets thicker with every reflow cycle. Wicking solder off the PCB or package pads does not remove IMC. Excessive IMC thickness can negatively affect solder joint reliability.

### 9.5.7 Sample Rework Parameters

Reworked part type >	BGAs (and other array area packages)	Processor Sockets
Process parameters based on	Sn/4Ag/0.5Cu and Sn/3.5Ag ball alloys. PCB nominal thickness 0.062-0.093" (1.6-2.4mm)	
Rework machine type	Hot air	
Flux applied to	Pads on board	
Solder paste	None	Same as used at SMT
Solder paste application	None	Printed on balls
Rising Ramp Rate below 205°C	0.5 - 2.5°C/ sec	
Soak Time, from 150°C to 215°C	≤ 100 sec (soak spec varies with solder paste selection)	
Critical Rising Ramp Rate between 205°C and 215°C	0.35 - 0.75°C/sec	
Peak Temperature Range	230-245°C	230-250°C
Time Above 217°C (TA217)	40-120 seconds	40-200 seconds
Delta-T (temp difference) across joints on part while above 217°C	≤10°C	≤15°C
Maximum Body Temp and Time	Not to exceed component supplier max specifications	
Except for body temps, all temperatures are measured at	At solder joints, with thermocouple in a hole at center of a pad.	
Falling Ramp Rate	0.5 - 2.0°C/ sec	

Here is a graphical representation of information in the table above.





### 9.5.8 Rework Profile Development

Because of the rework profile requirements, and because of the interactions that each profile adjustment makes, it can be very difficult to develop a rework profile. This is especially true if all stages of the profile are targeted for development simultaneously. A recommended approach is to break the profile down into phases, and develop the first phase first, then the second, and so on.

Here are some recommendations for successful rework profile development.

- Maximize bottom heater temperature when creating the profile.
- Keep the temperature of molten solder joints (above 217°C) within 10°C of each other across the component for BGAs, and within 15°C for sockets.
- Create the profile in steps. Don't move on to the next step until the current step meets goals. Developing the entire profile at once can be overwhelming.
  - Step 1: Board Preheat
    - Get joints into the 125 to 150°C range before lowering the nozzle.
  - Step 2: Soak
    - With nozzle down, get BGA joints into the 200 to 220°C range and socket joints into the 190 to 215°C range.
    - Check the Soak Time spec.
  - Step 3: Peak Reflow
    - With nozzle down, meet specs for peak reflow range and Time Above 217°C.
  - Step 4: Cool Down
    - With nozzle up, get board cool enough to handle safely.
- After developing the initial profile, check component body temperature to avoid exceeding Max component temps and times. Adjust profile as needed.
  - Place the body thermocouple at the topside location corresponding to the thermocouple with the hottest joint temperature.

Here is a graphical representation of the steps listed above.

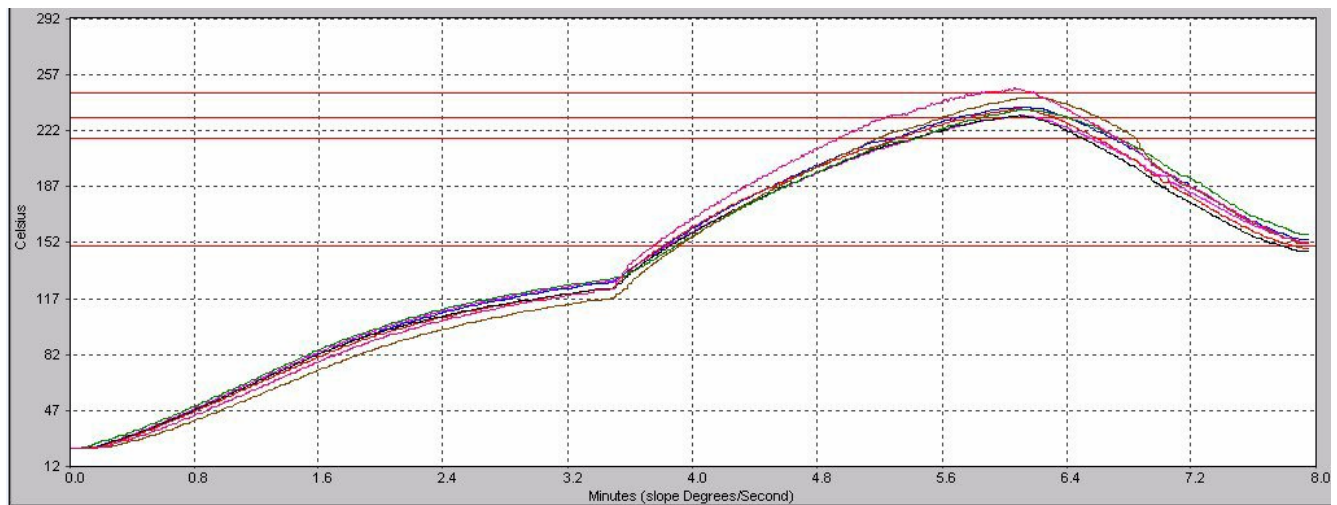




	Step 1 Board Preheat	Step 2 Soak, or FAT (Flux Activation Time)	Step 3 Peak Reflow	Step 4 Cool Down
	Start with solder joint temp $\leq 40^{\circ}\text{C}$ .  Preheat with bottom heater, before nozzle is lowered.	After nozzle is lowered, prior to peak reflow.	Nozzle is down during peak reflow.	Nozzle rises when joints go below $217^{\circ}\text{C}$ .
Target to exit this step	Solder Joint Temp: $125 - 150^{\circ}\text{C}$	BGA Solder Joint Temp: $200$ to $220^{\circ}\text{C}$ . Socket Solder Joint Temp: $190$ to $215^{\circ}\text{C}$ .	Peak Temp Range, and TA217 specs met.	Solder Joint Temp $\leq 80^{\circ}$ .
Other specs to check during this step	Rising Ramp Rate.	Rising Ramp Rate. Critical Rising Ramp Rate. Soak Time.	Peak Temp Range. TA217. Component Max Body Temp and Time.	Falling Ramp Rate.

## 9.5.9 Sample Rework Profiles

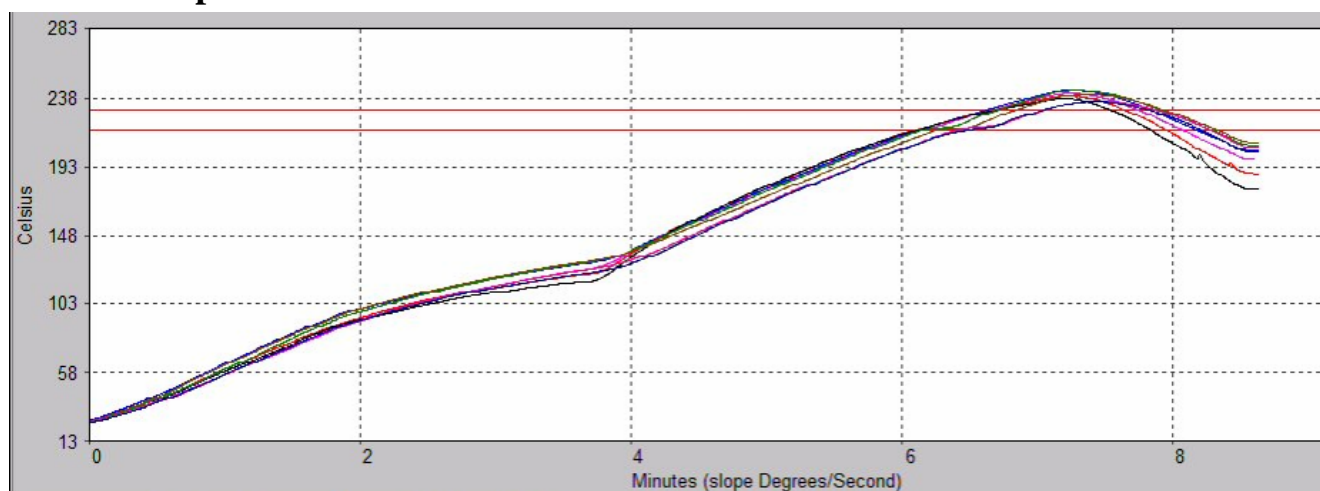
### 9.5.9.1 Sample BGA Rework Profile





	Peak	Max Rising Slope	Max Falling Slope	Rising Time Between 150/217	Total Time Above 217	Total Time Above 230	Total Time Above 245
1 Outer (1)	231.3	1.13	-1.02	95.00	68.51	12.37	0.00
2 Outer	237.1	1.21	-1.20	88.50	84.56	42.38	0.00
3 Outer	235.5	1.50	-1.18	89.88	77.67	34.33	0.00
4 Outer	231.3	1.36	-1.16	93.83	65.76	11.94	0.00
5 Center	235.0	1.00	-1.09	93.33	77.60	33.74	0.00
6 Top of Pkg	242.6	1.43	-1.84	76.57	99.58	60.67	0.00

### 9.5.9.2 Sample Socket Rework Profile



	Peak	Max Rising Slope	Max Falling Slope	Total Time Above 217	Total Time Above 230
1 A1 Out Cmr	240.7	0.78	-0.72	114.25	64.67
2 Out Cmr (CW)	242.5	0.72	-0.68	121.74	72.70
3 Out Cmr (CW)	239.1	0.96	-0.91	109.73	58.32
4 Out Cmr (CW)	237.4	1.11	-1.33	101.64	50.05
5 A1 In Cmr	242.5	0.71	-0.65	125.46	71.75
6 In Cmr (CW)	240.1	0.73	-0.64	120.50	65.66
7 In Cmr (CW)	235.5	0.69	-0.71	110.57	49.52
8 In Cmr (CW)	235.4	0.70	-0.75	101.15	46.51